

CURA 15.06 Printing Guidelines

CURA 15.06 software is supported by ULTIMAKER 3D printers.

Note: These are general printing guidelines. For detailed printing recommendations, please see the Printing Details tab of the 3D model.

To avoid printing problems, we recommend the following settings:

Quality

- Layer height: 0.1mm*
- Initial layer thickness: 0.3mm
- Bottom/top pattern: lines
- Skin perimeter line count: 0
- Horizontal expansion: 0.04mm (when entering 0.04mm index, the 3D printer perceives the model as monolith, i.e. as having no hidden layers).

*Layer Height 0.1mm is recommended for a better quality for the printed model. However, the printing time will double.

Speed setups:

- Infill speed: 50mm/s
- Outer shell speed : 30mm/s
- Inner shell speed: 20mm/s
- Top/bottom speed: 20mm/s
- Support speed: 50mm/s
- Travel speed: 150mm/s
- Skirt speed: 30mm/s
- Amount of slower layers: 4

Infill:

- Infill pattern: Grid
- Line distance: 3mm
- Infill overlap: 20%
- Infill layers: 0

Support:

- Enable support: Check
- Placement: Everywhere
- Overhang angle: 30°
- X/Y distance: 0.7mm
- Top distance: 0.2mm

- Bottom distance: 0.2mm
- Stair step height: 0.5mm
- Join distance: 0.7mm
- Area smoothing: 0.6mm
- Use towers: check
- Pattern: Grid
- Line distance: 1.5mm

Platform adhesion:

- Type: Brim
- Brim line count: 10 (up to 20, depending on the model's height)

Other 3D printing recommendations:

Outer Shell Speed - 20mm/s and Support Speed - 40mm/s settings apply to avoid model deformation. Apply other settings by default.

Disclaimer: This model will look outstanding if printed on SLA/SLS 3D printer. The accuracy of the model printed on FFF printer can vary from the result shown in the pictures.